International Journal of Mechanical and Production
Engineering Research and Development (IJMPERD)
ISSN (P): 2249-6890; ISSN (E): 2249-8001
Vol. 7, Issue 5, Oct 2017, 289-300


Original Article

# CFD APPROACH FOR OFF-DESIGN EFFICIENCY IMPROVEMENT OF DOUBLE SUCTION CENTRIFUGAL PUMP

**SHYAM NARAYAN SHUKLA[1], RUCHI KHARE[2] & VISHNU PRASAD[3]**

[1]*Corporate Research and Engineering Development, Kirloskar Brothers Limited, Pune, India*

[2,3]*Maulana Azad National Institute of Technology, Bhopal, India*

***ABSTRACT***

***The energy consumed by a large pump to transport the water has a dominant role in pumping system and hence the efficiency of the pump is of great concern to both manufacturer and customer in large size pumps. The pumps mostly operate at off-design conditions and hence at lower performance. A little improvement in the efficiency of pump will save huge amount of energy and money. In order to improve performance of double suction centrifugal pump, especially at off-design conditions, systematic and validated computational fluid dynamics (CFD) approach had been used to assess the flow behavior in suction casing, impeller and delivery casing of an existing pump for identification of geometrical locations for performance improvement. The head losses and flow pattern in the individual part of the pump were studied and identified the locations for performance improvement and accordingly, the geometry of pump components and surface roughness were modified. There was an increase of normalized head and efficiency by 4.5% and 3.1% respectively at duty point as compared to the CFD results of existing pump. The substantial increase in head and efficiency was also achieved at off-design conditions for both existing and smooth surfaces.***

***KEY WORDS: Double Suction, Horizontal Split Case, Suction Casing, Delivery Casing & Pump Efficiency***

**Received:** Aug 22, 2017**; Accepted:** Sep 14, 2017**; Published:** Oct 03, 2017**; Paper Id.:** IJMPERDOCT201730

## INTRODUCTION

The double-suction centrifugal pumps are used for conveyance of a large quantity of water through long pipeline and have wide application due to its centrifugal structure with large flow rate, stable running operation and convenient maintenance.

In double suction centrifugal pump, the curved flow passage with two suction intake and presence of rotating blades of impeller make the flow very complex. The designer has to make some assumptions to carry out the design and hence it becomes necessary to predict the pump performance. Computational Fluid Dynamics (CFD) has emerged as a cost effective tool for flow simulation in pumps for obtaining detailed flow behavior and derivation of pump performance. It has minimized the conventional physical pump model tests. The pressure and velocity distribution give a good insight of fluid flow behavior within the flow passage and helps to identify the defects in pump geometry. The Reynolds Averaged Navier Stokes (RANS) equations are solved using numerical techniques like the finite control volume method to obtain pressure and velocity distributions in the flow domain using CFD [Shah S.R. et al. (2013)].

The numerous commercial CFD codes are used for flow simulation. The accuracy of numerical flow simulation of commercial software depends on many parameters and selection of these parameters requires experience and the validation of CFD results with physical model test. CFD has been used for design improvement

of pump by changing the geometry and number of the impeller vanes. [Chakraborty et al. (2012), Bacharoudis et al. (2008) Houlin et al. (2010), Rajmane and Kallurkar (2015), Leng et al. (2013)]. Numerous investigators have worked to improve performance (Head-Efficiency) at design point [Yumiko Sekino et al. (2016)].

In the present work, the flow simulation in existing double suction horizontal split casing centrifugal pump has been carried out to obtain flow behavior and performance characteristics using commercial CFD code Ansys CFX. The computed head and efficiency from CFD are compared with those derived from experimental results to validate the flow simulation. Based on the flow behavior and loss characteristics, the locations for geometry improvement are identified and then geometries of pump components are modified with the goal to improve the efficiency of pump both at design and off-design conditions. Surface roughness also plays a significant role in the performance of the pump and hence, in addition to geometry modification, the flow simulation is extended to study the effect of roughness on performance of pump with smooth walls in this paper.

## PUMP GEOMETRY AND BOUNDARY CONDITIONS

The geometry of double suction horizontally split case pump consists of double suction, impeller with 5 vanes and a volute delivery casing. The design head and discharge coefficient of pump are 0.433 and 0.223 respectively, with specific speed NSUS 1710. Out of these three components, impeller is rotating and the other two (suction casing and delivery casing) are stationary. Geometric modeling and meshing of all components is done together and interface is given to segregate each domain. The assembled pump geometry for flow simulation is shown in figure1.

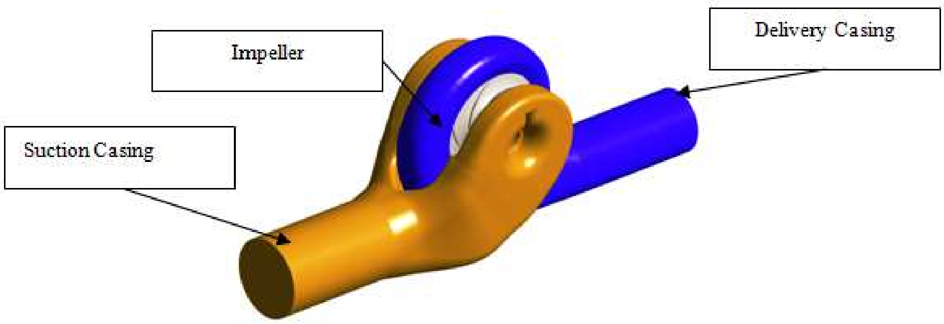


**Figure 1: Pump Geometry**

The geometry is created using Pro Engineer solid modeler. The assembly of pump geometry is imported to ANSYS ICEM CFD16 for geometry cleaning and meshing.

The unstructured hybrid tetrahedral mesh is generated in flow domain and fine prismatic layers near the surface of impeller vanes for proper resolution of the boundary layer. The appropriateness of mesh size has been fixed by mesh independence study. There are 300,000, 1,120,000 and 200,000 nodes in suction casing, impeller and delivery casing respectively.

The outer enclosing surfaces of all components are taken as walls and no slip condition with appropriate wall roughness is given. The modified geometry is also checked for hydro dynamically smooth walls. In this case, the walls are taken as smooth walls. As the velocity distribution at inlet is unknown and hence total pressure at suction inlet and mass flow rate at delivering casing outlet are specified. The interface between rotating and stationary components is taken as a frozen rotor. The rotational speed of the impeller is specified in a rotating frame of reference, while suction and volute

regions are set as stationary. The flow is assumed to be steady in a rotating frame of reference. The water is considered as incompressible with constant density and viscosity. The scalable Log-law is used near wall. First flow simulation is done using different turbulence models and based on a comparison of results with experimental values, Standard κ - ε turbulence model is used for further simulations [Shukla et al. (2016)]. The high resolution advection scheme is used for solution.

## COMPUTATION OF PARAMETERS

The numerical simulation gives values of pressure and velocity distribution, in the flow passage of the pump. The following local and global parameters are computed for performance assessment:

Head developed $$H = \frac{TP_O - TP_I}{\gamma} \quad (1)$$

Discharge coefficient $$Q_s = \frac{Q}{\omega r_2^3} \quad (2)$$

Head coefficient $$\psi = \frac{gH}{\omega^2 r_2^{\ 2}} \quad (3)$$

Total head loss $$H_{HL} = H_S + H_I + H_D \quad (4)$$

Relative head loss in suction $$\xi_S = \frac{H_S}{H + H_{HL}} * 100 \quad (5)$$

Relative head loss in impeller $$\xi_I = \frac{H_I}{H + H_{HL}} * 100 \quad (6)$$

Relative head loss in casing $$\xi_C = \frac{H_D}{H + H_{HL}} * 100 \quad (7)$$

Relative total head loss $$\xi_{HL} = \frac{H_{HL}}{H + H_{HL}} x100 \quad (8)$$

Hydraulic efficiency $$\eta_{hyd} = \frac{H}{H + H_{HL}} * 100 \quad (9)$$

Overall efficiency $$\eta_o = \eta_{hyd} * \eta_{vol} * \eta_{mech} \quad (10)$$

## PERFORMANCE ANALYSIS OF EXISTING PUMP

The numerical flow simulation in existing is carried out for seven discharge values varying from 2% to 120% of design discharge at constant rotational speed of the pump. The values of the discharge coefficient corresponding to 2%, 20%, 40%, 60%, 80%, 100% and 120% are 0.004, 0.045, 0.089, 0.134, 0.179, 0.223 and 0.268 respectively. The flow simulation is carried out from 2% discharge value because the simulation at zero discharge is not possible.

In experimental results, the measured overall efficiency includes volumetric and mechanical efficiencies and these

are not estimated separately in the experiment. It is also not possible to estimate these losses from CFD analysis. Hence the volumetric and mechanical efficiencies corresponding to the specific speed and discharge at rated conditions are estimated from the graphs given by Karassik [IX], and assumed to be same at all discharges for computation of overall efficiency. The computed local and global parameters of the flow simulation results are presented in qualitative and quantitative form. The quantitative values of global parameters, i.e. head and discharge coefficient and efficiency obtained from numerical flow simulation are compared with the experimental data to validate the suitability of numerical scheme for this pump.

To calculate the hydraulic efficiency of the pump from simulation results, the hydraulic losses are estimated in suction casing, impeller and volute casing. These losses combined together form the basis for hydraulic efficiency of the pump.

The numerical flow simulation is conducted using following turbulence models:

- κ-ε model
- Shear stress transport (SST)
- Renormalized group κ-ε (RNG κ-ε)
- κ-ω turbulence model

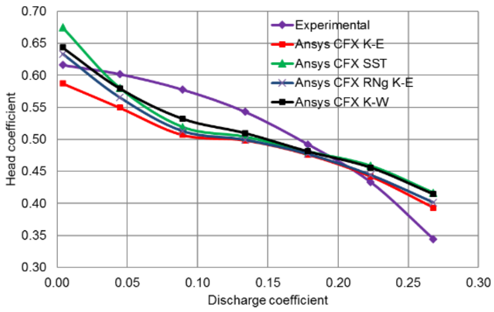


**Figure 2: Variation of Head**

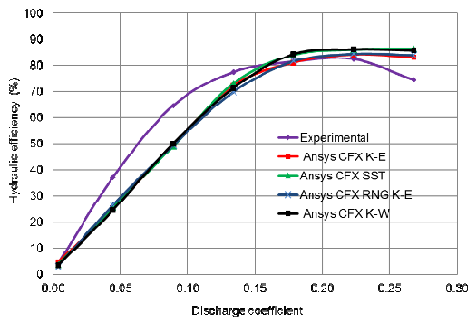


**Figure 3: Variation of Hydraulic Efficiency**

It is seen from figure 2 and figure 3 that variation patterns of head and hydraulic efficiency and their values near design condition are closely matching with experimental results but there are large variations at off design condition. It is observed from the figure3 that for 20-80% load conditions, head and efficiency obtained from CFD are less than the experimental value while at overload i.e. 120%, their values from CFD are more than experimental. This indicates that the losses are not properly accounted in CFD analysis at off-design conditions, inability to capture losses due to secondary

flows. The hydraulic efficiency is nearly same up to 60% discharge for all turbulence models, but Standard κ–ε model gives efficiency close to the experimental value near the design point as seen from figure 3.

The hydraulic losses are computed in each component of the pump from CFD results for different discharge and normalized with the total head developed, including losses using equations 5, 6, 7 and 8. There is a drastic drop in loss as discharge increases from 2% to 60%. The negative value of the loss in suction at 2% discharge in figure 4 indicates that at very low discharge, there can be a lot of variations in loss prediction by CFD in suction passage due to highly circulatory flow at suction. It is seen from figure 4 that RNG κ-ε estimates the slightly higher loss in suction than Standard κ – ε model.

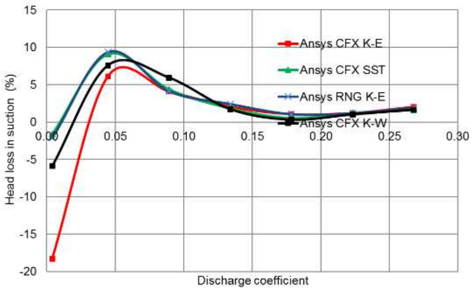


**Figure 4: Variation of Loss in Suction**

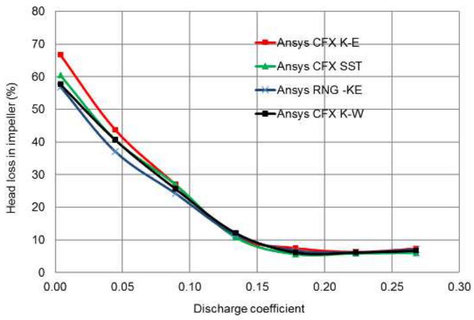


**Figure 5: Variation of Loss in Impeller**

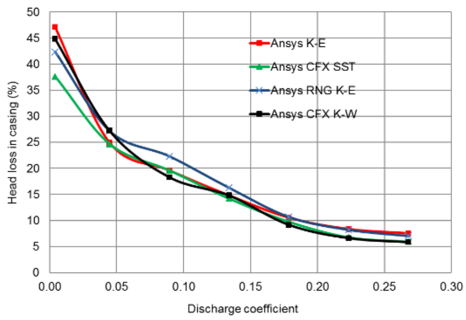


**Figure 6: Variation of Loss in Casing**

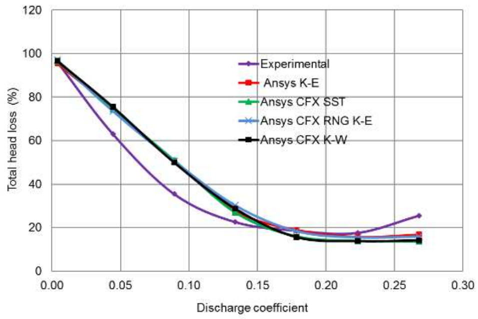


**Figure 7: Variation of Total Head Loss in Pump**

The loss variation of impeller in figure 5 predicts that, loss decreases as the discharge increases and becomes minimum at design discharge, and then becomes nearly constant. The prediction impeller loss is more with κ – ε as compared to other models.

The loss in casing is also found to decrease with an increase in discharge in figure 6. It is seen that κ–ε gives less loss at lower discharge and more loss at higher discharge, while loss prediction by κ-ω model is opposite to κ–ε model.

The total loss obtained from different turbulence models is compared with the experimental values in figure7. It is observed that total loss obtained in experiments is lower than that obtained from CFD for 2% to 80% load while it is more at overload condition and hence it is just reverse of the hydraulic efficiency chart. The total loss variation up to 80% load condition from different turbulence models is nearly same, but loss values obtained from different models differ after 80% load. The minimum loss is predicted by κ-ω model. The total loss value closely matches with the experimental value for RNG κ – ε and κ – ε models near the design discharge.

The variations of hydraulic losses in different components and total loss indicate that the losses are more at the lower discharge and decrease as the discharge increases up to design point. The losses obtained in design discharge are nearly same for all turbulence models in different components.

The deviation of results at off-design conditions triggered to conduct flow simulation with transient condition. Since κ–ε turbulence model has close matching with experimental results and consumed less time for solution, it is used for transient simulation. The appropriateness of time scale is checked by carrying out simulation with different time steps similar to mesh independent study.

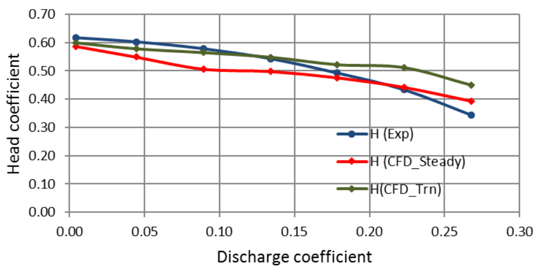


**Figure 8: Variation of Head in Transient and Steady State**

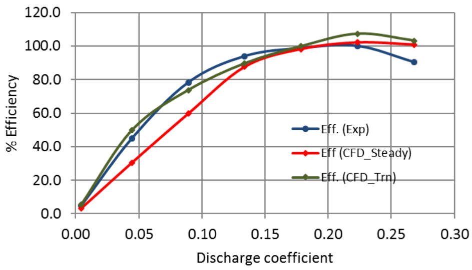


**Figure 9: Variation of Efficiency in Transient and Steady State**

As seen from figure 8 and figure 9 that the deviation of the head and discharge coefficient from experimental at part load conditions is reduced but deviated at duty point. The results at duty point closely match with experimental results and hence study state simulation is adopted.

## GEOMETRY MODIFICATIONS

The flow simulation of existing pump gave an insight of fluid flow pattern throughout the passage of the pump. The improvement in performance is attempted by modifying the geometry of suction casing, delivery casing and impeller based on loss and flow pattern obtained from simulation results. The unsymmetrical nature of flow is seen in suction casing and hence there is a possibility of improvement of performance by suitably modifying the suction casing. The theoretical comparison with existing delivery casing gave an idea to increase the base circle diameter. This has resulted in the improved performance of the pump. Secondly, it is perceived from flow patterns that present location of baffle plate is obstructing the water flow when water is entering into the impeller at eye location and needs to be shifted. There is the large recirculation zone near the baffle plate at upstream side.

Modification in suction casing has resulted in a significant loss reduction when simulation is conducted with modified suction casing. The original and modified geometry of suction casing is shown in figure 10.

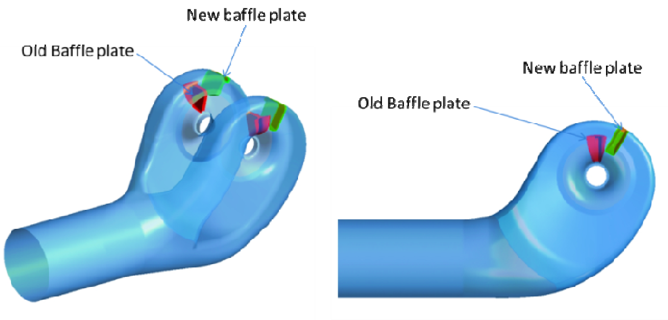


**Figure 10: Geometry Modification of Suction Casing**

In case of delivery casing, the disturbances are seen at the interfaces and hydraulic losses in the delivery casing are also found to be on the higher side. It is observed that the base circle diameter to impeller outlet diameter is not proper as per guidelines given in the different pump design books. Therefore, the base circle diameter is increased and checked the performance of pump with increased base circle diameter. It is found that with modified delivery casing, the flow became smooth and has resulted in loss reduction. The comparison of original and modified delivery casing is shown in figure 11.

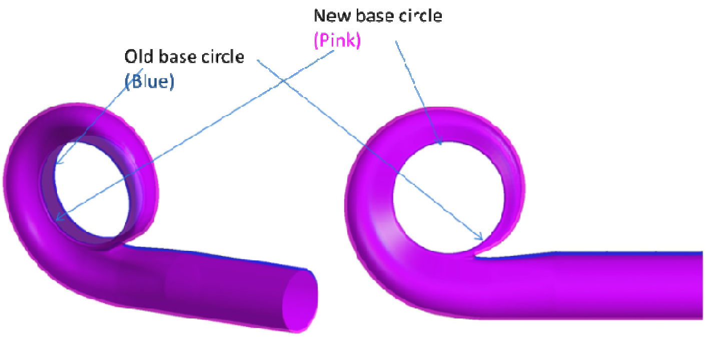


**Figure 11: Geometry Modification of Delivery Casing**

After having encouraging results with geometry modifications of suction and delivery casing, the following modifications in impeller geometry have been attempted:

- Smoothing of impeller area and inlet vane angle.
- Reduced meridional length at hub side and smooth variation of vane angle.
- Changed tip angle at shroud keeping same meridional length.
- Reduced impeller eye diameter as per pump hand book and taken smooth area variation in impeller.
- Best impeller profile selected from data bank and scaled down to suit present pump.

No significant improvement in pump performance could be achieved with modifications of impeller geometry. Therefore, the final pump design consists of modified suction casing, original impeller and modified delivery casing.

## PERFORMANCE ANALYSIS OF MODIFIED PUMP

The flow simulation in pump with modified geometry was done to assess its performance. Generally, variation of pump load is 20% of design conditions and hence the results for 80%, 100% and 120% were compared with existing pump in Table-1.

**Table 1: Comparison of Normalized H-Q and Eff.-Q Performance Improvement**

| Description | % Flow Rate | % Head | % Efficiency | % Head Increase | % Eff. Increase |
|---|---|---|---|---|---|
| Experimental | 80 | 113.6 | 99.0 | - | - |
| | 100 | 100.0 | 100.0 | - | - |
| | 120 | 79.5 | 90.4 | - | - |
| CFD Analysis (Original) | 80 | 109.9 | 98.2 | - | - |
| | 100 | 102.0 | 102.2 | - | - |
| | 120 | 90.8 | 100.9 | - | - |
| CFD Analysis (Modified) | 80 | 113.1 | 100.3 | 3.2 | 2.1 |
| | 100 | 106.5 | 105.3 | 4.5 | 3.1 |
| | 120 | 96.1 | 104.8 | 5.3 | 3.9 |
| CFD Analysis (Modified-Smooth) | 80 | 115.5 | 108.4 | 5.6 | 10.1 |
| | 100 | 111.0 | 110.2 | 9.0 | 8.0 |
| | 120 | 103.0 | 110.7 | 12.2 | 9.8 |

The values of the head and efficiency from CFD were normalized with the corresponding value of head and efficiency obtained from experimental results at duty point for comparison. It was seen from Table-1 that there was

increase in normalized head by 4.5% and normalized efficiency by 3.1% at duty point as compared to results of existing pump. For smooth surface, increase in head and efficiency at duty point was found to be 9.0% and 8.0% respectively. It was also observed that there was a significant increase in head and efficiency at off-design conditions. The improvement in the head was found to increase with discharge in modified pump. The increase in efficiency with discharge was found to be 2.1% to 3.9% for discharge 80% to 120% for rough surface. In case of smooth surface, its improvement was found to be less i.e. 8% at duty point than 80% and 120% discharge i.e. 10.1% and 9.8% respectively.

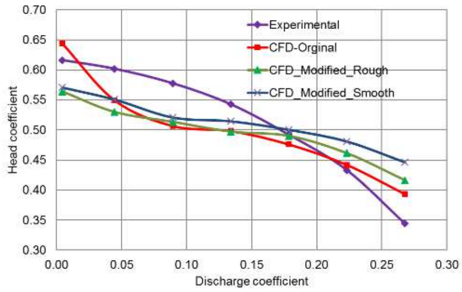


**Figure 12: Variation of Head**

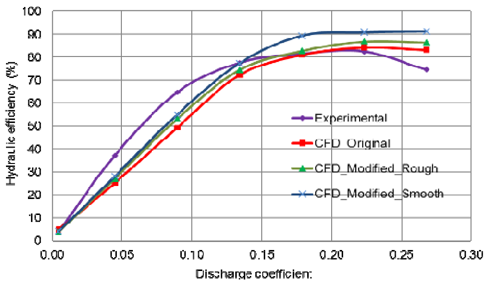


**Figure 13: Variation of Hydraulic Efficiency**

The best efficiency point for smooth surface was shifted to right side, i.e. 120% of the flow. It means that the effect of smoothening of the surface had increased water carrying capacity of the pump. The variation of efficiency curve had been flat after 80% in the case of smooth surface.

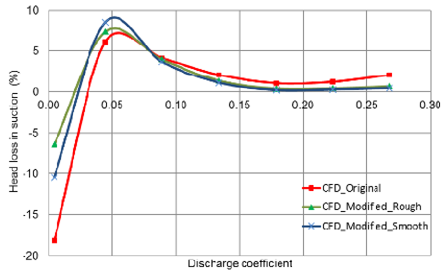


**Figure 14: Variation OF Head Loss in Suction**

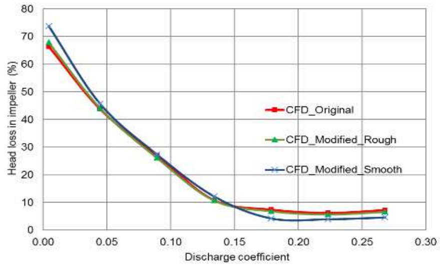


**Figure 15: Variation of Head Loss in Impeller**

The pattern of loss in suction, impeller and delivery casing shown figure 14 to figure 16 indicates the losses decrease as the discharge increases. The computed losses in all three components were found to be less in modified pump than the existing pump for load 40% to 120%. At small flow rate, loss prediction by CFD can vary a lot in suction casing. This may be due to highly circulatory flow at suction. There was a drastic drop in loss as discharge increases from zero to 60% in impeller and casing.

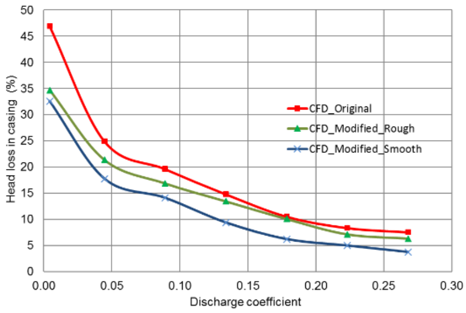


**Figure 16: Variation of Head Loss in Casing**

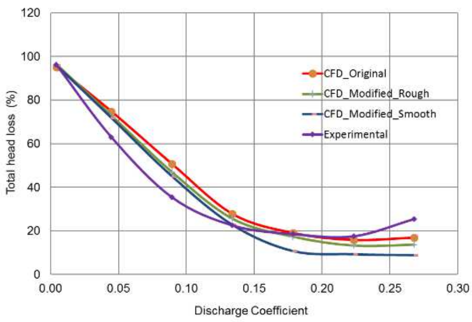


**Figure 17: Variation of Total Head Loss in Pump**

The total loss in modified pump was less than the existing pump as shown in figure 17. It was observed that total loss obtained in the experiment was much lower than that obtained from CFD for 2% to 60% load while it was more at overload condition and which and hence it was just reverse of hydraulic efficiency variation.

## CONCLUSIONS

The head and efficiency predictions of double suction horizontal split casing pump closely match with experimental values near duty point and deviates at off-design conditions. The significant reduction in losses and improvement in the head and efficiency at off-design conditions is achieved after geometry modification in suction and delivery casing of existing pump. The performance due to smoothening of surface is found to be greatly enhanced. The

modified pump performance gave better performance for a wide range of pump operation near duty point.

## NOMENCLATURE

$H_S$ - head loss across suction casing (m)

$H_I$ - head loss across impeller of pump (m)

$H_D$ - head loss across delivery casing (m)

$Q$ - Discharge of pump ($m^3/s$)

$g$ - Gravitational acceleration ($m/s^2$)

$T_{PI}$ - total pressure at impeller inlet (Pa)

$T_{PO}$ - total pressure at impeller outlet (Pa)

$r_2$ - pump impeller outlet radius (m)

$\omega$ - Rotational speed of impeller (rad/s)

$\gamma$ - Specific weight of water ($N/m^3$)

$\eta_{pump}$ - pump efficiency (%)

$\eta_{hyd}$ - hydraulic efficiency (%)

$\eta_{vol}$ - volumetric efficiency (%)

$\eta_{mech}$ - mechanical efficiency (%)

## *REFERENCES*

1. *Shah S.R., Jain S.V., Patel R.N. and Lakheria V.J., (2013), "CFD for centrifugal Pump: A review of the state of the art" Procedia Engineering 51, 2013, pp 715 – 720.*
2. *ChakrabortySujoy, Pandey K.M., RoyaBidesh, (2012), "Numerical Analysis on Effects of Blade Number Variations on Performance of Centrifugal Pumps with Various Rotational Speeds", International Journal of Current Engineering and Technology, Volume 2, No.1, pp 143-152.*
3. *E.C. Bacharoudis, A.E. Filios, M.D. Mentzos and D.P. Margaris, (2008), "Parametric Study of a Centrifugal Pump Impeller by Varying the Outlet Blade Angle". The Open Mechanical Engineering Journal, 2, pp75-83.*
4. *Houlin. LIU, WANG Yong, YUAN Shouqi, TAN Minggao and WANG Kai, (2010), "Effects of blade number on characteristics of centrifugal pump", Chinese Journal of Mechanical Engineering, Volume 23, pp1-6.*
5. *RajmaneSatish M, S. P. Kallurkar, (2015)," Enhancing Performance of Domestic Centrifugal Pump by Use of CFD Tool by Changing Design Parameters", International Journal of Informative & Futuristic Research, Volume 3, Issue 2 October, pp 579-590.*
6. *Veeralkumar Thakur, Tarun Yadav & Rajiv B, CFD Assisted Nozzle Design in Burr Removal Process, International Journal of Mechanical and Production Engineering Research and Development (IJMPERD), Volume 6, Issue 2, March - April 2016, pp. 23-36.*

7. *Leng, H. F.; Wang, F. J.; Zhang, Z. C.; Yao, Z. F.; Zhou, P. J.(2013), "Suppression of secondary flows in a double suction centrifugal pump with different loading distributions", Materials Science and Engineering, Volume 52, Issue 3, 2013, pp1-6.*
8. *ANSYS-CFX Users guide, V 15, by M/s ANSYS Inc. USA, 2015.*
9. *Shukla Shyam Narayan, Khare Ruchi, Prasad Vishnu, (2016), "Performance evaluation of turbulence models for flow simulation of double suction centrifugal pump", International Journal of Civil Engineering and Technology (IJCIET), Volume 7, Issue 6, pp 1-10.*
10. *Karassik Igor J, Messina Joseph P, Cooper Paul, Heald Charles C., "Pump Hand book" Fourth Edition, the McGraw-Hill Publication, 2008.*
11. *Yumiko Skein, Hiroshima Watanabe, (2016), "Design optimization of double suction volute pumps", Elsevier's World pumps, Volume? 2016, Issue 5, 2016, pp 32-35.*